CHEMICAL ENGINEERING TRANSACTIONS

VOL. 45, 2015

A publication of AIDIC

The Italian Association of Chemical Engineering
www.aidic.it/cet

Guest Editors: Petar Sabev Varbanov, Jiří Jaromír Klemeš, Sharifah Rafidah Wan Alwi, Jun Yow Yong, Xia Liu

ISBN 978-88-95608-36-5; **ISSN** 2283-9216 DOI: 10.3303/CET1545199

Validation of Developed Modified Plug-Flow Furnace Model for Identification of Burner Thermal Behaviour

Zdeněk Jegla, Jan Horsák, Vojtěch Turek*, Bohuslav Kilkovský, Jiří Tichý

Institute of Process and Environmental Engineering, Faculty of Mechanical Engineering, Brno University of Technology, Technická 2, 616 69 Brno, Czech Republic
turek@fme.vutbr.cz

The paper discusses results of the next stage of development of the so-called Modified Plug-Flow Furnace Model (MPF) which was in its initial formulation originally presented two years ago. The MPF model is intended for calculation of heat flux distribution from a burner along the height (length) of a cylindrical combustion chamber through identification of axial distribution of heat released rate from a burner.
Results of a careful and complex validation are presented via comparisons of model data and experimental data obtained during tests of a large set of process burners of various types and duties, operated under diversified combustion conditions including several distinct fuels being combusted. The validation confirms high accuracy of the developed MPF and thus makes it possible to directly identify for a given burner the resulting axial distribution of the rate of released heat (that is, the axial fuel burnt profile of a burner necessary for accurate heat flux distribution calculation).
This encouraging observation gives an opportunity to utilise the validated MPF model in future work in the course of development of an accurate practical method for thermal design of process equipment employing combustion chambers such as power boilers, fired heaters, furnaces, etc.

1. Introduction

Combustion chambers containing inbuilt tubular heat transfer systems - such as power boilers, fired heaters for refinery, chemical or petrochemical industry, waste incinerators in waste-to-energy applications, heat transfer fluids heaters in pharmaceutical or food-processing industry, etc. – usually serve as the primary hot utilities in mentioned industrial applications and unambiguously belong to the most energy consuming equipment dominantly influencing economy of these plants.
Energy saving design and operation of this equipment together with its operating reliability and required lifetime can be achieved only by proper application of integration techniques considering not only systematic approach for proper placement and conceptual design of such equipment but also its reliable detail design based on accurate thermal-hydraulic calculations (Lam et al., 2014).
While the various available calculating tools for systematic approach (Varghese and Bandyopadhyay, 2012), conceptual technical-economic analysis of placement (Tucker and Ward, 2012) and basic sizing of this equipment (Jegla, 2006) are on considerable high level, practical calculating tools for detailed thermal-hydraulic design of combustion equipment such as boilers and furnaces, however, fall far short of the necessary accuracy (Jegla et al., 2010).
This article addresses the area of the detailed design of the aforementioned combustion equipment. Its purpose is to contribute to its more accurate thermal design by the means of taking into account the actual thermal behaviour of industrial burners during the design process.
Computational modelling is an important tool for identification of the thermal-hydraulic behaviour of the fired heaters and boilers, mainly due to the fact that experiments performed on operated industrial boilers and furnaces can be realised only in a very limited scale. While cases of existing equipment under operation can be checked or analysed by detailed computational techniques, for example based on computational fluid dynamics (CFD), as was done in the case of the furnace section of a pulverized coal-fired boiler by Al-Abbas and Naser (2013) or in the case of the radiant section of a refinery fired heater by

Please cite this article as: Jegla Z., Horsák J., Turek V., Kilkovský B., Tichý J., 2015, Validation of developed modified plug-flow furnace model for identification of burner thermal behaviour, Chemical Engineering Transactions, 45, 1189-1194 DOI:10.3303/CET1545199

Jegla et al. (2014), these computationally-intensive approaches cannot be used during the design stage of new combustion equipment. The main reason is unacceptably long time required to obtain results, but also the fact, that these sophisticated approaches are capable of checking given design only, but they are not capable of sizing procedures required in the detail design stage. For these reasons, it is possible or even necessary to use some empirical simplified global calculation tools in the design stage of this equipment. These calculation tools, however, are generally not able to sufficiently take into account some of the important detail circumstances (Bahadori and Vuthaluru, 2010), i.e. the use of design computational tools of insufficient quality might lead to not receiving the equipment design that is economical and/or exhibiting high efficiency and long lifetime (Jegla et al., 2011).

2. Characteristics of the developed MPF model

The knowledge of the real distribution of heat flux in a combustion chamber is one of the most important factors affecting the quality of its design. Current design methods are based on the use of average heat flux, which, however, does not provide sufficient information needed for exact design of inbuilt tubular heat transfer systems, especially when two-phase (i.e. vapour-liquid) flow of heated medium inside tubes occurs (Jegla et al., 2011). Avoidance of these thermal design problems can be achieved by the developed modified plug-flow (MPF) model for identification of burner thermal behaviour. The MPF model was first formulated by Jegla (2013) for a given specific burners´ testing combustion chamber and this paper is devoted to detailed verification and validation of the model's behaviour.

As the name suggests, the MPF model is principally based on the plug-flow (or long-furnace) model, described for example by Hewitt (1994). Following division of the combustion chamber into several zones using energy balance equations and iterative calculations the mean temperature of the zone gas volume (containing flame, unburnt fuel and combustion air) is determined and the transferred heat is then calculated. Modification of the original model provided by Jegla (2013) ensured that the model behaviour became more consistent with the observed behaviour of the actual burning process.

The principle of the MPF model is schematically presented in Figure 1, on the example of horizontally oriented cylindrical combustion chamber of arbitrary length *L* (specific dimensions of the actual combustion chamber used for model validation are presented in chapter 3.1 of this paper). Its shell is divided into *n* number of small water-cooled length segments. Local heat flux from the hot gases to the heat sink of *i-th* segment (q_i, [W/m^2]) is calculated from the heat (Q_i, [W]) absorbed by the segment heat transfer area (A_i, [m^2]). Therefore, the MPF model is able (based on comparison with the results of burner combustion tests) to provide the heat flux distribution profile along the combustion chamber length and to identify corresponding profile of local rate of heat release per unit length of the chamber typical for the given tested burner and combustion conditions.

3. Validation of the MPF model

The ability of the developed MPF model to operate successfully, its general properties, applicability on real combustion processes and accuracy were all validated by the means of its application on a series of combustion tests primarily focused on identifying thermal characteristics of various burners. These burners combustion tests took place at the experimental burner testing facility of the Institute of Process and Environmental Engineering (IPEE) of the Faculty of Mechanical Engineering (FME), Brno University of Technology (BUT). Brief description of the facility follows.

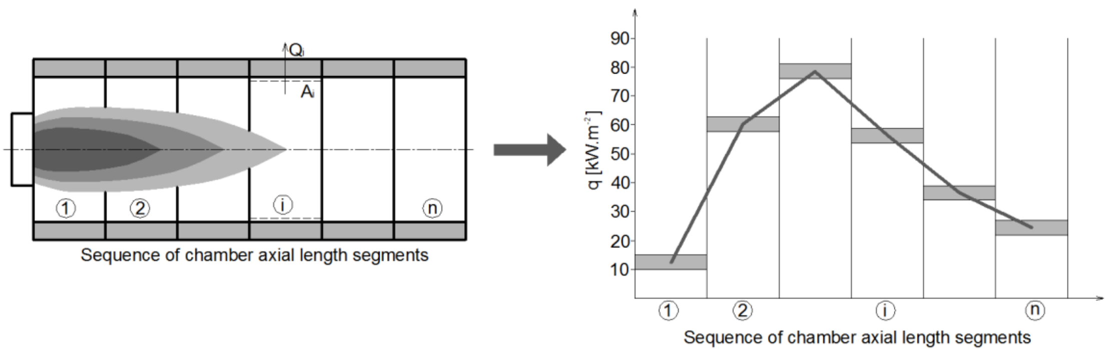

Figure 1: Principle of the MPF model for identification of heat flux along the combustion chamber length

3.1 Experimental facility description

The primary purpose of the IPEE experimental burner testing facility is testing, research and development of industrial burners. The centrepiece of the facility is a horizontal double-shell water-cooled combustion chamber of overall length 4 m with 1 m internal diameter. A single burner is situated at one front of the chamber, while stack entrance is situated at the other, both fronts are equipped with insulating lining. Remaining inner surfaces of the chamber are plain and non-insulated. What makes the combustion chamber special and particularly suitable for MPF model validation is its unique construction comprising of division of the inter-shell space into the total of seven separated sections (0.5 m long each, with the exception of the first section with 0.4 m and the last section with 0.9 m), each of them with an independent supply of cooling water. Such construction enables a relatively simple determination of the heat flux absorbed along the length of the chamber from cooling water inlet and outlet temperatures and its flow rate, which were all measured during tests. Further information about the layout of the chamber and its measuring equipment can be found in Kermes and Bělohradský (2013) or in greater detail in Vondál (2012).

3.2 Scope of validation

The data that were subject to comparison were primarily heat flux values along the length of the combustion chamber. As indicated above, average specific heat flux values for each section were obtained (calculated) from measured values of cooling water inlet and outlet temperatures and its flow rate. Data were collected in 2 minute intervals during the whole test, however only values obtained in the period when the burning process was stabilised were considered relevant. The relevant time span usually was between 22 and 26 minutes (11-13 measured values), while at some cases this value was considerably lower (or higher). Some other measured values than those mentioned above, namely the values of fuel inlet flow rate, combustion air temperature and flue gas oxygen concentration (used to determine the inlet volumetric flow rate of combustion air) were supplied to the model and served as its input data. The MPF model's most crucial input data were, however, expected amounts (expressed by fraction) of provided fuel that burned out in each individual section. These values were at first estimated and subsequently altered several times (i.e. iteratively specified) in order to obtain the heat flux profile that was as much in accordance with the measured one as possible. The heat flux profile calculated from the measured data was, however, a subject to measuring equipment errors; therefore reaching a 100 % match between both profiles did not necessarily mean an actual 100 % accuracy of the model. Applying the error propagation theory (Bělohradský, 2010) resulted in evaluation of heat flux errors in the following Case study 1 in the interval between approximately 0.70 kW/m^2 and 1.40 kW/m^2, or between 2.30 % and 4.90 % in relative terms. Errors in other cases are not expected to differ significantly.

The ability of the developed MPF model to successfully simulate the heat flux profile suggested by the experimental data was assessed by its application on the total of 22 different combustion cases in which various fuels, oxidisers, burner heat outputs and ambient conditions were used. Oxygen-enriched combustion air was used in 8 of these cases, while various liquid fuels were used in 6 of them. The following subsection presents results obtained from 3 typical cases, one of each category, i.e. one with conventional fuel and ambient combustion air, one with conventional fuel and oxygen-enriched combustion air and one with a liquid fuel. The overall accuracy of the model from the heat flux match point of view was evaluated as good, very good or even excellent as is illustrated by the Table 1.

3.3 Case study 1

The first presented case deals with somewhat standard burning conditions, i.e. natural gas-fired low NOx burner with a thermal duty of 751 kW supplied with ambient air. Introduction of the oxidiser (air) to the flame was realised in two stages. Heat flux profiles obtained from the MPF model and CFD simulations were compared with the profile based on the measured data and the result is shown in Figure 2. An apparent shift of the maximum heat flux farther from the burner compared to usually published shape of combustion heat flux profiles was caused by a relatively great length of the flame in comparison to the overall combustion chamber length due to staged air supply effect.

Several computational sub models have been used during CFD simulations (see Bělohradský, 2010) and the CFD heat flux profile shown in Figure 2 represents results that were most in accordance with the measured data of all the CFD simulations. It is obvious that while the MPF model provides fairly good approximation of the real heat flux profile (reaching maximum deviation of approximately 1.45 kW/m^2 or 4.90 %), even the best available CFD results are far less accurate while being far more time demanding at the same time. Presented results were achieved for the following set of burned-out fuel fractions:

Table 1: Average and maximum deviations of heat flux values provided by the MPF model from measured values

Categories of tested cases	Average deviation, all sections, all cases	Single maximum deviation	Average of maximum deviations for all cases
Conventional fuel (natural gas), ambient combustion air (8 cases)	1.38 %	5.12 %	3.50 %
Conventional fuel (natural gas), oxygen-enriched combustion air (8 cases)	1.92 %	8.48 %	4.65 %
Liquid fuels, ambient combustion air (6 cases)	2.31 %	11.76 %	6.28 %

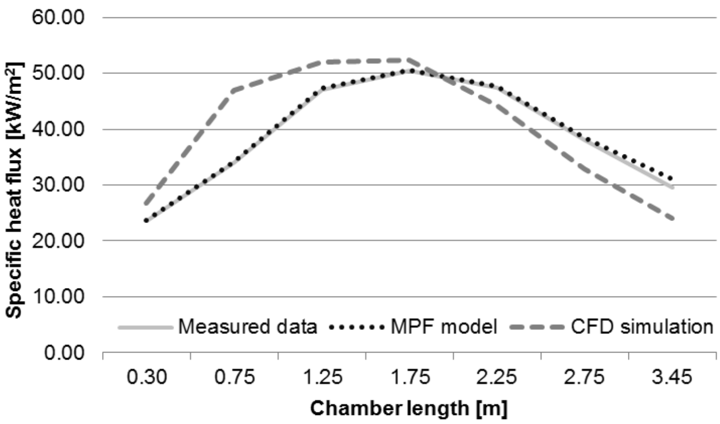

Figure 2: Comparison of heat flux profiles obtained from the MPF model, CFD simulations and experimental measurement for Case study 1

Table 2: Fractions of the total amount of fuel burned in each section of the experimental combustion chamber in case 1 according to the MPF model

Section of the chamber	1	2	3	4	5	6	7
Burned fuel [% vol./100]	0.462	0.197	0.114	0.112	0.062	0.023	0.030

3.4 Case study 2

Case study 2 illustrates the ability of the MPF model to successfully simulate low-NO_x burner with performance-augmenting oxygen-enriched combustion. In this case, the extra oxygen was provided to the combustion air prior to its contact with the fuel (which was natural gas supplied to the burner in two stages), reaching the value of 33.2 % (oxygen in air). Thermal duty of the selected burner was 749 kW. CFD simulations of this case were not performed, so only MPF model-provided and measured heat flux profiles are shown on Figure 3.

The highest discrepancy between the two heat flux profiles reaches approximately 2.10 kW/m^2 in absolute terms and 5.15 % in relative terms. Both values can be considered low, suggesting very good accuracy of the MPF model. Corresponding fractions of fuel burned in each section according to the model are presented in Table 3.

higher peak flux in general is caused by oxygen-enrichment of the air), which is, aside from Figure 3, indicated also by the MPF model by the means of considerably increased value of the amount of burned The two-staged fuel combustion provides significantly different heat flux profile than the one achieved in

Table 3: MPF model-provided data of the amount of fuel burned in each section of the experimental combustion chamber in case 2

Section of the chamber	1	2	3	4	5	6	7
Burned fuel [% vol./100]	0.399	0.307	0.115	0.092	0.027	0.027	0.033

case 1 for approximately the same thermal duty. Peak heat flux values are much closer to the burner (the fuel in section 2 of the chamber. For further information about oxygen-enhanced combustion tests on this particular combustion chamber including overall performance evaluation and observed NOx formation behaviour see Bělohradský et al. (2014).

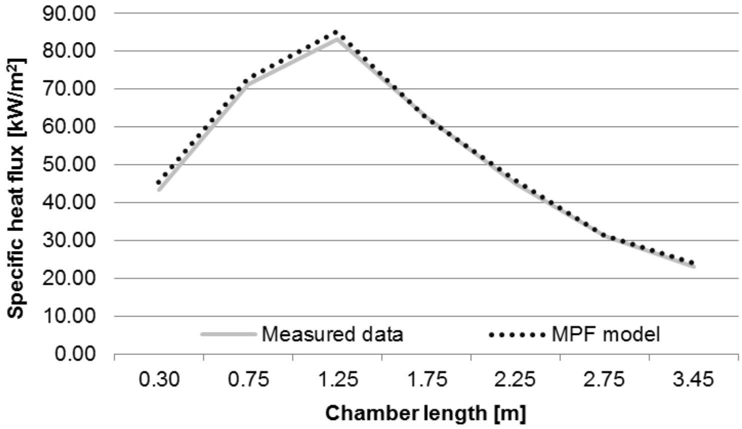

Figure 3: Comparison of heat flux profiles obtained from the MPF model and experimental measurement for Case study 2Case study 3

The final case presented in this paper addresses the combustion of an alternative liquid fuel, specifically the methyl-ester of rapeseed oil (RME). Non-preheated combustion air was used as the atomising medium and the total thermal duty of the burner reached 900 kW. As well as in Case study 2 above, no CFD simulations were performed, so only measured and MPF model-based data were available for comparison which is shown on the following Figure 4.

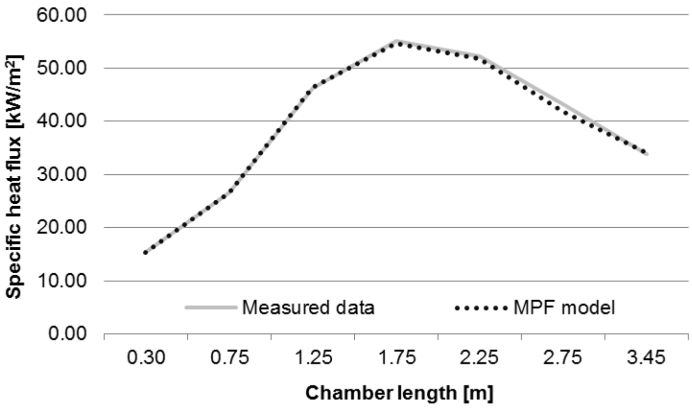

Figure 4: Comparison of heat flux profiles obtained from the MPF model and experimental measurement for Case study 3

Obtained results clearly show that even simulation of the combustion of liquid fuels is possible with great accuracy using the MPF model. The greatest observed difference between measured and MPF-provided heat flux profiles is approximately 1.40 kW/m^2 or 3.20 % with the setting described in the following Table 4.

Table 4: Fractions of the total amount of fuel that were burned in individual sections of the combustion chamber suggested by the MPF model in case 3

Section of the chamber	1	2	3	4	5	6	7
Burned fuel [% wt./100]	0.442	0.246	0.159	0.112	0.041	0.000	0.000

The peak heat flux is shifted even farther from the burner than in case 1 due to increased thermal duty and longer flame due to two-staged combustion air input. Further information about some liquid fuels combustion tests (including RME), some of which were simulated using the MPF model, including experimental settings, fuel properties and overall results can be found in Kermes and Bělohradský (2013).

4. Conclusions

This paper presents the results of performed validation of the developed MPF model for identification of the thermal behaviour of burners. Results of the three selected case studies representing three different categories of the total of 22 analysed burners with various combustion conditions clearly show, that the presented validation of the developed MPF model can be considered as successful, i.e. that the model is capable of providing the real heat flux distribution profile along the combustion chamber length and identifying the corresponding profile of local rate of heat release per unit length of chamber representing thermal behaviour of a given burner at specific combustion conditions.

The results of the MPF model thus can be used as a reliable input data for the next stage of development which purpose is to develop a reliable and detailed thermal calculation method suitable for detail design of combustion chambers containing inbuilt tubular heat transfer systems (such as those in power boilers, fired heaters or heat transfer fluids heaters).

Acknowledgement

The authors gratefully acknowledge financial support provided by Technology Agency of the Czech Republic within the research project No. TE02000236 “Waste-to-Energy (WTE) Competence Centre”.

References

Al-Abbas A.H., Naser J., 2013, Computational fluid dynamic modelling of a 550 MW tangentially-fired furnace under different operating conditions, Procedia Engineering, 56, 387-392

Bahadori A., Vuthaluru H. B., 2010, Novel predictive tools for design of radiant and convective sections of direct fired heaters, Applied Energy, 87, 2194-2202.

Bělohradský P., 2010, Methods for Determination of Characteristic Parameters of Combustion Processes on the Basis of Experiments and Modeling. Brno University of Technology, Brno, Czech Republic (in Czech).

Bělohradský P., Skryja P., Hudák I., 2014, Experimental Study of Oxygen-Enhanced Combustion on NOx Emission, In-Flame Temperatures and Heat Flux Distribution, Chemical Engineering Transactions, 39, 787-792.

Hewitt G., Shires A.L., Reg Bott T., 1994, Process Heat Transfer. Boca Raton: CRC Press.

Jegla Z., 2006, The Conceptual Design of Radiant Chamber and Preliminary Optimization of a Process Tubular Furnace, Heat Transfer Eng., 27, 50-57.

Jegla Z., 2013, Development of modified plug-flow furnace model for identification of burner thermal behavior, Chemical Engineering Transactions, 35, 1195-1200.

Jegla Z., Bébar L., Pavlas M., Kropáč J., Stehlík P., 2010, Secondary Combustion Chamber with Inbuilt Heat Transfer Area - Thermal Model for Improved Waste-to-Energy Systems Modelling, Chemical Engineering Transactions, 21(2), 859 – 864.

Jegla Z., Hájek J., Vondál J., 2014, Numerical analysis of heat transfer in radiant section of fired heater with realistic imperfect geometry of tube coil, Chemical Engineering Transactions, 39, 889-894.

Jegla Z., Kohoutek J., Stehlík P., 2011, Design and operating aspects influencing fouling inside radiant coils of fired heaters operated in crude oil distillation plants, Proceedings of International Conference on Heat Exchanger Fouling and Cleaning - 2011, June 05-11, Crete Island, Greece, <www.heatexchanger-fouling.com/papers/papers2011/2_Jegla_F.pdf> accessed 21.02.2015

Kermes V., Bělohradský P., 2013, Biodiesel (EN 14213) heating oil substitution potential for petroleum based light heating oil in a 1 MW stationary combustion facility, Biomass and Bioenergy, 49, 10-21.

Lam H.L., Varbanov P.S., Klemeš J.J, 2014, Applied thermal engineering towards sustainable development, Applied Thermal Engineering, 70, 1051-1055.

Tucker R., Ward J., 2012, Identifying and quantifying energy savings on fired plant using low cost modelling techniques, Applied Energy, 89, 127-132.

Varghese J, Bandyopadhyay S., 2012, Fired heater integration into total site and multiple fired heater targeting, Applied Thermal Engineering, 42, 111-118.

Vondál J., 2012, Computational Modeling of Turbulent Swirling Diffusion Flames, PhD Thesis, Brno University of Technology, Brno, Czech Republic.